1. Use both LT and National Instruments SPICE (Multisim) to plot the voltage transfer characteristic $V_o$ versus $V_i$ for the circuit shown below for $-10 \text{ V} \leq V_i \leq 10 \text{ V}$. Assume that the op amps are ideal and that each of the diodes is a 1N4148 (the SPICE parameters for the diode are saturation current $1 \text{nA}$, emission coefficient 1.8, and breakdown voltage $100 \text{ V}$). The resistor values are $R = 7.3 \text{k}\Omega$. Compare the simulation results with the theoretically expected values with regard to break points and slopes. Also plot the currents in the diodes as functions of $V_i$.

2. Use both LT and National Instruments SPICE (Multisim) to plot the output voltage $v_o(t)$ as a function of time for the circuit shown below if the input is

$$v_i(t) = A \sin(\omega t)$$

where $A = 3 \text{ V}$ and $f = 1 \text{ kHz}$ for two cycles of $v_i(t)$. Compare the peak value of the output with the theoretically expected value.